



**THESEUS FE**

## **Vehicle thermal safety with THESEUS-FE**

Author: Dr. Daniel Koester

Date: September 4, 2014

## ANSA or other CAD/meshing software

- **CAD cleanup and meshing:** either CAD geometry description or NASTRAN-style FE meshes

## THESEUS-FE

- **Thermal model building:** density, specific heat capacity, and conductivity of all vehicle parts, emissivity of important surfaces

## Star-CCM+

- **CFD model building:** air mass flow and temperature from fan and heat exchanger and other inlets

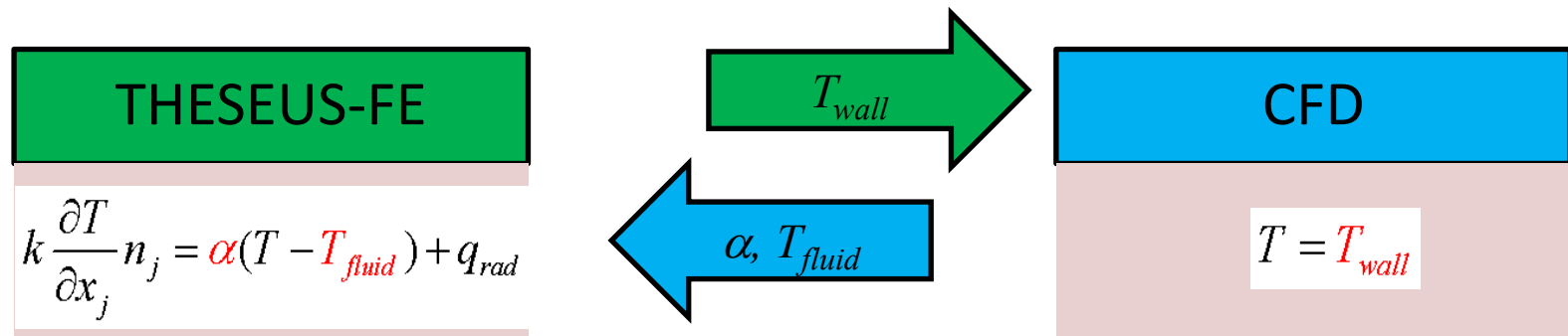
## THESEUS-FE Coupler

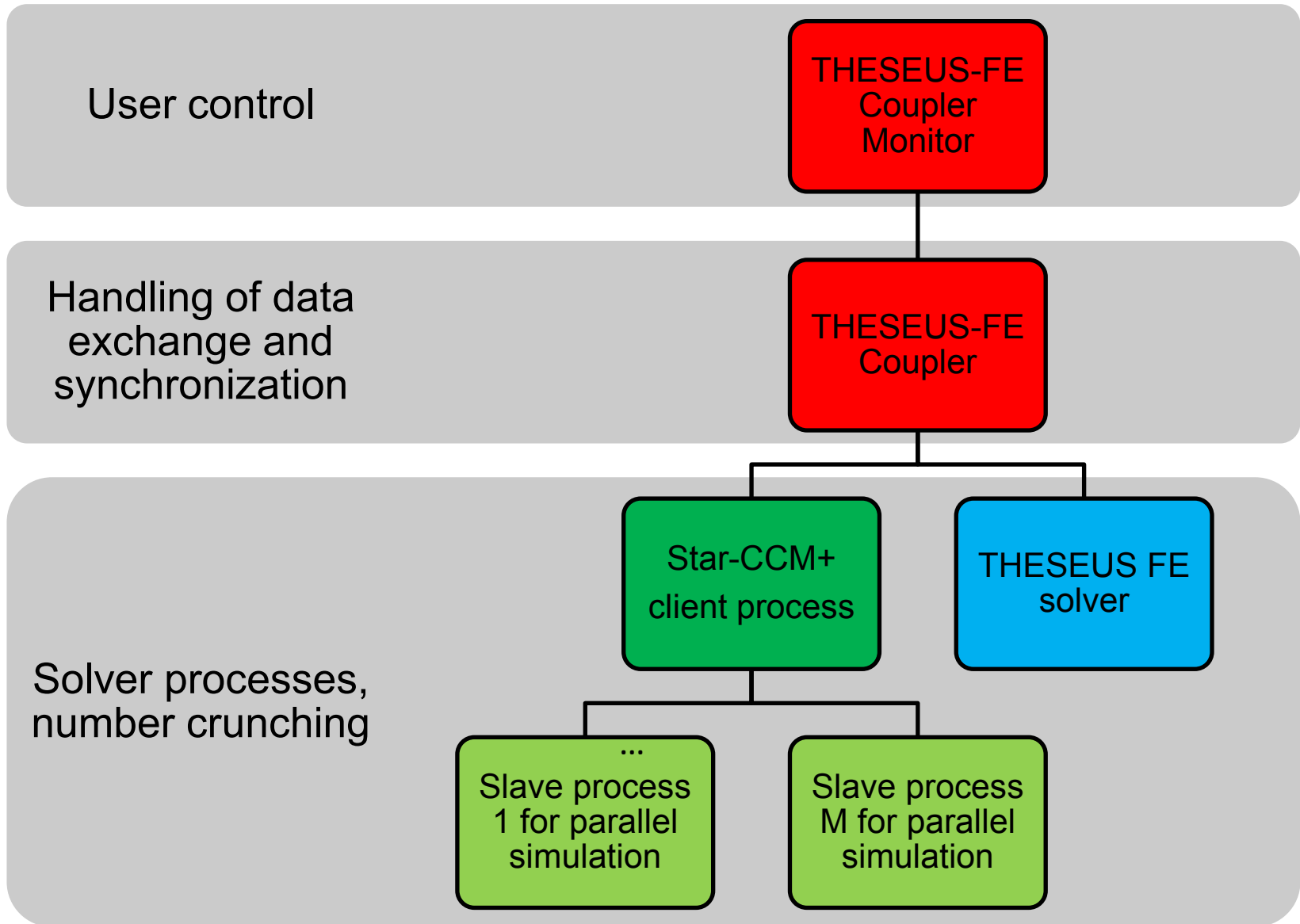
- **Coupled simulation setup and analysis:** identification of coupled interfaces between solid parts and air (engine, gearbox, heat shields, etc.)

## THESEUS-FE or other postprocessing software

- Identification of hot spots or heat flow balance

The interaction between individual solvers is done through the boundary conditions:

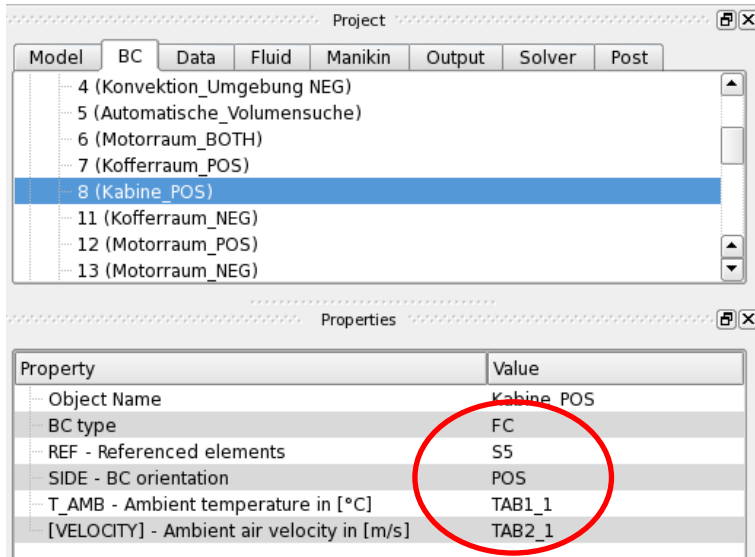




- ✓ Create THESEUS-FE and CFD model.
- ✓ Test both models in “standalone modus”, running independently:  
THESEUS-FE can use dummy convection parameters, CFD can use dummy wall temperatures  
Any convergence problems with these independent test models must be dealt with now – co-simulation will not work otherwise!
- ✓ Open THESEUS-FE case in the GUI. Start the Coupler Setup Dialog.  
Define communication mode (file-based or via TCP/IP)
- ✓ Start solver processes and coupler process (in any order). Coupler process will take control of the solver processes
- ✓ Start the Coupler Monitor in the THESEUS-FE GUI to control the running co-simulation task.
- ✓ Post-processing .
- ✓ Possibly more simulations



- Replace any AIRZONE objects – convection in fluid regions will now be treated by the CFD code.
- BC objects describing the convection between solid parts and fluid regions need to be changed:



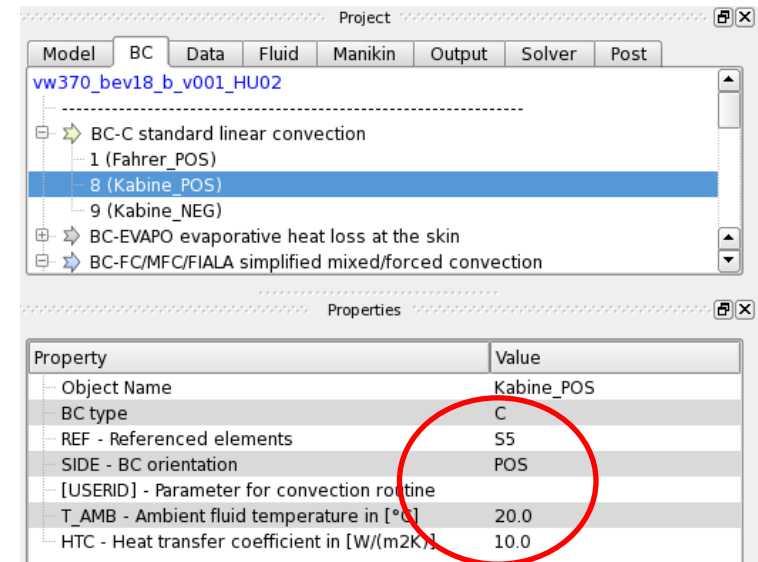
Project

Model BC Data Fluid Manikin Output Solver Post

- 4 (Konvektion\_Umgebung NEG)
- 5 (Automatische\_Volumensuche)
- 6 (Motorraum\_BOTH)
- 7 (Kofferraum\_POS)
- 8 (Kabine\_POS)
- 11 (Kofferraum\_NEG)
- 12 (Motorraum\_POS)
- 13 (Motorraum\_NEG)

Properties

Property	Value
Object Name	Kabine_POS
BC type	FC
REF - Referenced elements	S5
SIDE - BC orientation	POS
T_AMB - Ambient temperature in [°C]	TAB1_1
[VELOCITY] - Ambient air velocity in [m/s]	TAB2_1



Project

Model BC Data Fluid Manikin Output Solver Post

vw370\_bev18\_b\_v001\_HU02

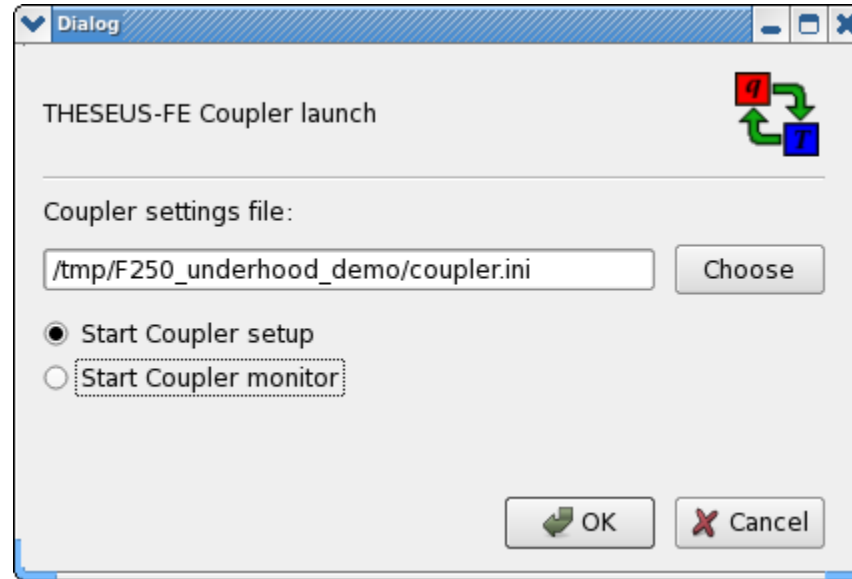
- BC-C standard linear convection
  - 1 (Fahrer\_POS)
  - 8 (Kabine\_POS)
  - 9 (Kabine\_NEG)
- BC-EVAPO evaporative heat loss at the skin
- BC-FC/MFC/FIALA simplified mixed/forced convection

Properties

Property	Value
Object Name	Kabine_POS
BC type	C
REF - Referenced elements	S5
SIDE - BC orientation	POS
[USERID] - Parameter for convection routine	
T_AMB - Ambient fluid temperature in [°C]	20.0
HTC - Heat transfer coefficient in [W/(m2K)]	10.0

- An initial condition with CONVT=TEMPE avoids extremely large heat fluxes at the beginning of the simulation.

- Load the standalone THESEUS-FE case into GUI.
- The Coupler launch dialog will guide the user through all necessary preparation steps:



- First select option “Start Coupler Setup”.
- The Coupler working directory will later contain a file „coupler.ini“ with all selection coupling options. It is a good idea to put the THESEUS-FE and CFD solver files in sub-directories of the working directory.

# Coupler Setup Dialog Page 1

- Page 1 of Coupler Setup Dialog: global configuration

Select Star-CCM+  
and the .sim file

Settings for  
TCP/IP-  
communication:  
Where will the  
Coupler process  
run, and how will  
you connect to it?

THESEUS-FE Coupler setup  
Global settings

Define global settings for the THESEUS-FE Coupler here. Right-click on the entries in the left column to open help. If "Next" button is disabled, leave the mouse pointer over it to get feedback about missing or incorrect data.

Couple THESEUS-FE case with:  StarCCM+

Flowmaster (three-way AC coupling)

OpenFOAM

Coupler network settings: Coupler will run on this host:

Use this port for incoming messages from THESEUS-FE GUI:

Use this port for incoming messages from solver processes:

Identification token:

First coupling time for transient simulations

< Back    Next >    Cancel



# Coupler Setup Dialog Page 2

- Page 2 of Coupler Setup Dialog: exchanged quantities:

THESEUS-FE sends wall temperature

Star-CCM+ sends  $\alpha$  und  $T_{\text{fluid}}$

Number of coupling interfaces, see next page

THESEUS-FE Coupler setup

THESEUS-FE Coupler setup  
Coupling setup between THESEUS-FE and 3D-CFD

THESEUS-FE will send to StarCCM+:

Variable	Initial value	Units	Description
<input checked="" type="checkbox"/> TEMPE	20	C	Wall temperature
<input type="checkbox"/> MNDMX	0	kg/m <sup>2</sup> -s	Manikin steam production rate

StarCCM+ will send to THESEUS-FE:

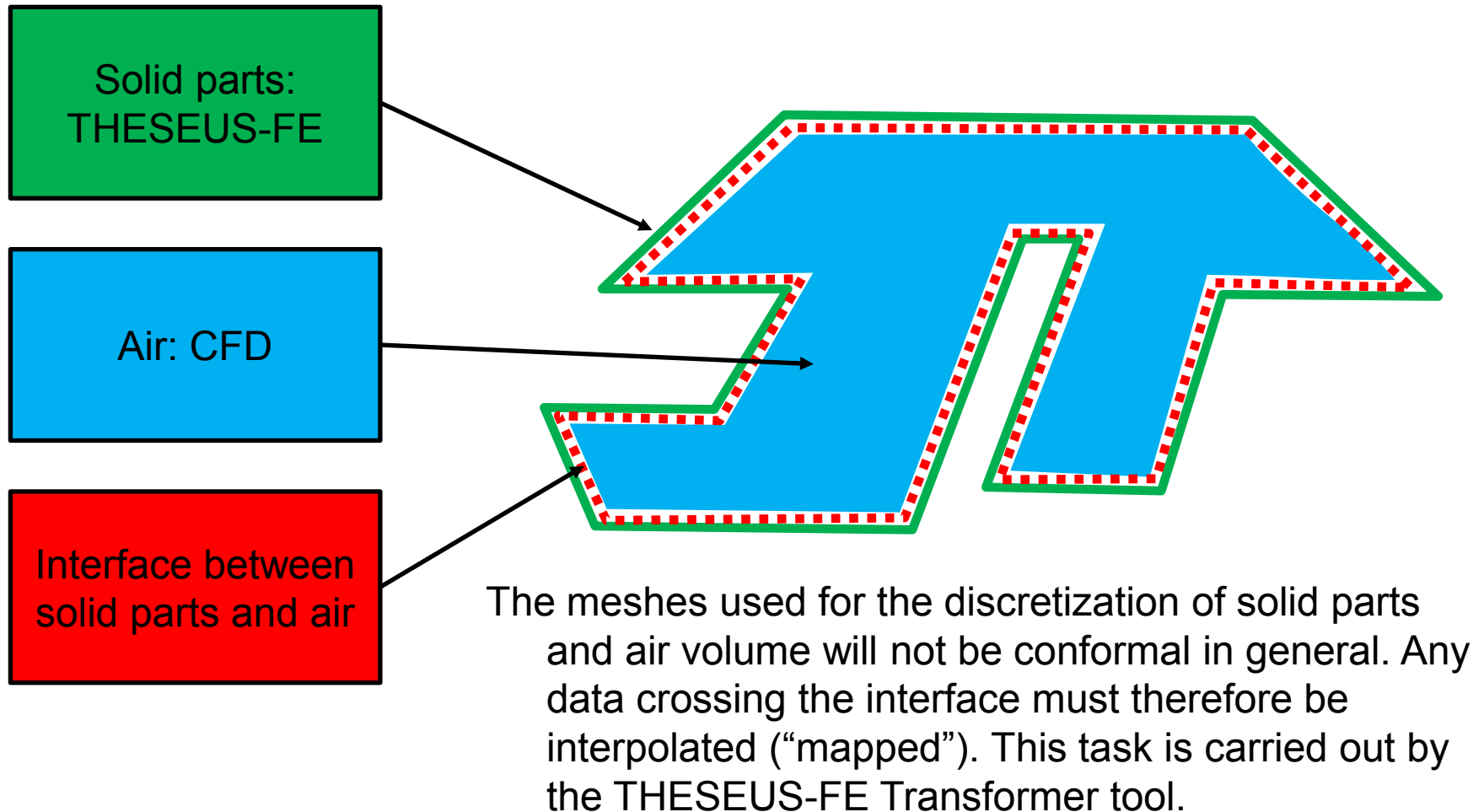
Variable	Initial value	Units	Description
<input checked="" type="checkbox"/> CONVC	10	W/m <sup>2</sup> -K	Local heat transfer coefficient
<input checked="" type="checkbox"/> CONVT	20	C	Local fluid film temperature
<input type="checkbox"/> MFH2O	0.015	1	Ambient steam mass fraction
<input type="checkbox"/> PRESS	101300	Pa	Ambient pressure

Number of coupling interfaces: 2

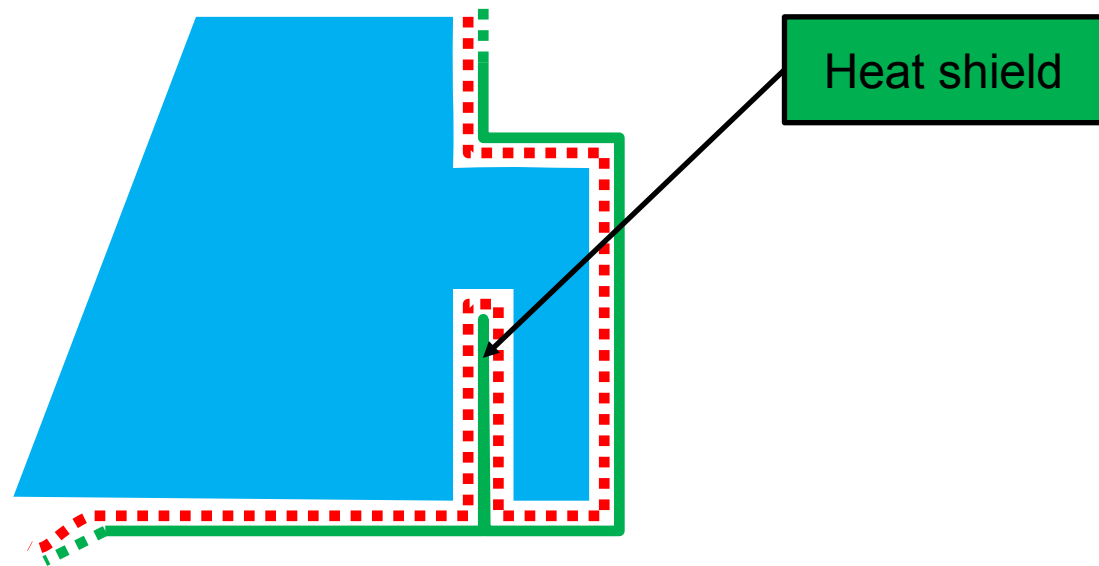
< Back   Next >   Cancel



An **Interface** is the boundary between the physical domains treated by the individual solvers.

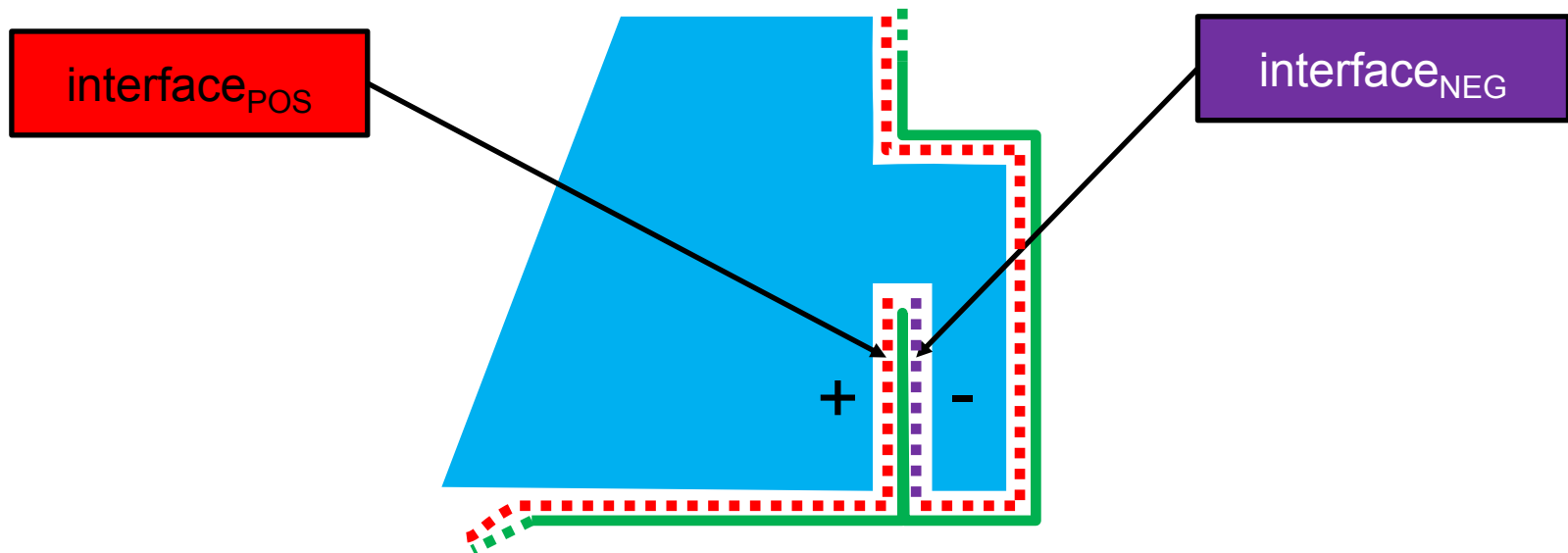


Thin-wall parts, e.g. a heat shield, will usually be modelled using a shell layer in THESEUS-FE. Both sides of the wall are in contact with air:



With „naive“ mapping of  $\alpha$  und  $T_{\text{fluid}}$  one risks incorrect results since the Transformer cannot distinguish between front and back of a shell surface – front and back shell surface are at the same geometric position. The mapped  $T_{\text{fluid}}$  field will be a random mix of front and back values.

Our solution for this problem: the user must define two separate interfaces. Data mapping is carried out independently for each interface:



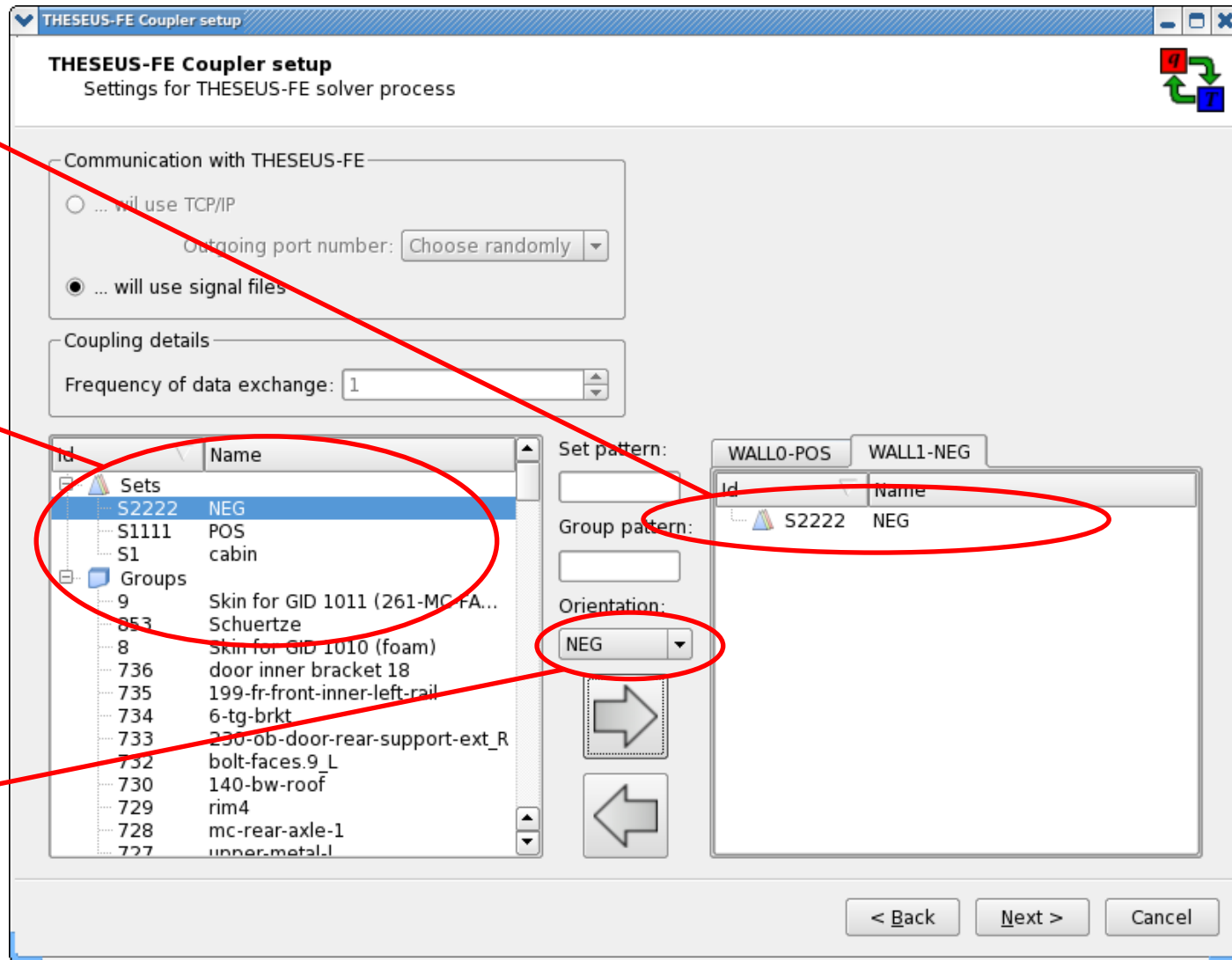
interface<sub>POS</sub> and interface<sub>NEG</sub> are assigned to the positive respective negative shell side in THESEUS-FE. The Transformer will now map convection values from adjoining fluid volume elements to the correct shell side.

Page 3 of the Coupler Setup Dialog: interface definition for THESEUS-FE:

One of more SETs and/or PIDs can be used to define the interface

All SETs and PIDs occurring in the case

Select the correct orientation for the interface here!



# Coupler Setup Dialog Page 4

Page 4 of the Coupler Setup Dialog: Interface definition for Star-CCM+:

This button starts a Java macro to analyze the .sim file. Click this first!

Use the pattern selection helper to select all Star-CCM+ boundaries whose name starts with "POS"

Repeat this step for negative interface

The screenshot shows the 'THESEUS-FE Coupler setup' dialog box. The 'Communication with StarCCM+' section has the radio button '... will use TCP/IP' selected. The 'Case preparation using Java macro' section has 'Adapter.java' as the macro name and a path to the StarCCM+ executable. A green button labeled 'Run macro to analyze case' is highlighted. The 'Coupling details' section shows a frequency of data exchange of 1. The 'Pattern' field is set to 'POS\*'. The 'WALL0-POS' and 'WALL1-NEG' lists are visible, with 'WALL1-NEG' circled in red. The 'Run macro to analyze case' button is also circled in red.

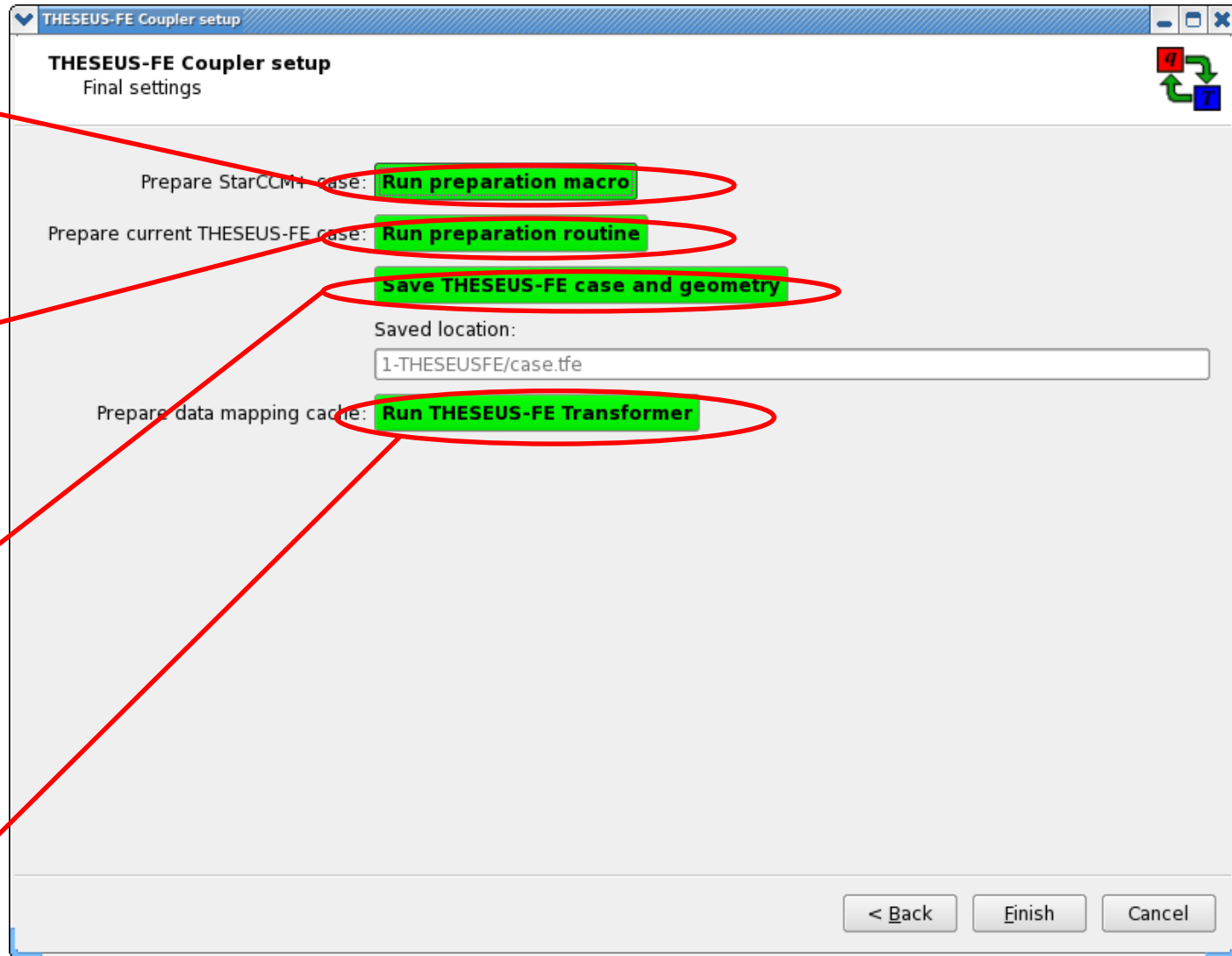
## Page 5 of Coupler Setup Dialog: Concluding steps

Finalize Star-CCM+ settings

Finalize THESEUS-FE settings

Save THESEUS-FE case

Create interpolation matrix for Transformer



The Coupler monitor is used to control the co-simulation.

Coupler controls

Log window

